

ISSN : 2321-9416



Indo - American Journal of Mechanical Engineering



www.iajme.org

Email : iajme.editor@gmail.com or editor@iajme.org

Predicting the Axial Steam Turbine's Performance using 3D Computational Fluid Dynamics

Rajesh and Mohan

Department of Mechanical Engineering, Visakha Technical Campus,
Narava, Visakhapatnam, Andhra Pradesh, India

Article Info

Received: 30-12-2024

Revised: 17-01-2025

Accepted: 29-01-2025

ABSTRACT

To generate electricity, steam turbines are the primary prime movers. With horsepower ratings exceeding 10,000 HP, these turbines discover significant uses for naval propulsion in addition to electricity production. We use Ansys-CFX CFD software to model and analyze the performance of a single axial turbine stage, which has 58 blades in the rotor and 60 blades in the stator. The working fluid for the turbine stage was determined to be superheated steam with temperatures of 500 K and 600 K at 2 & 3 pressure, respectively. The working fluid's thermodynamic characteristics are sourced from the IAPWS97 database. When designing turbo equipment, it is essential to first do thorough simulation studies to identify potential issues, and then determine the best operating conditions. Under both inlet situations, the turbine is operated at various speeds, varying between 6,000 and 12,000 rpm. At a greater pressure of 3 bar, the power output more than doubles, but the efficiencies remain almost the same, exceeding 90%, regardless of the input circumstances. Maximum efficiency were achieved at 8000 rpm at 2 pressure and 10000 rpm at 3 bar, which ranged by 15 to 20%. Stator blades exhibit a significant rise in kinetic energy, suggesting a highly efficient nozzle. Due to the fact that the relative mach number is dependent on the rotor speed and since the sonic velocity decreases as temperatures fall, it is seen that both the temperatures and the relative mach number drop as the speed increases.

Key words: Prime movers, Ansys, CFX, CFD, IAPWS97, Turbo Machinery, Nozzle Efficiency, Kinetic Energy.

1. INTRODUCTION

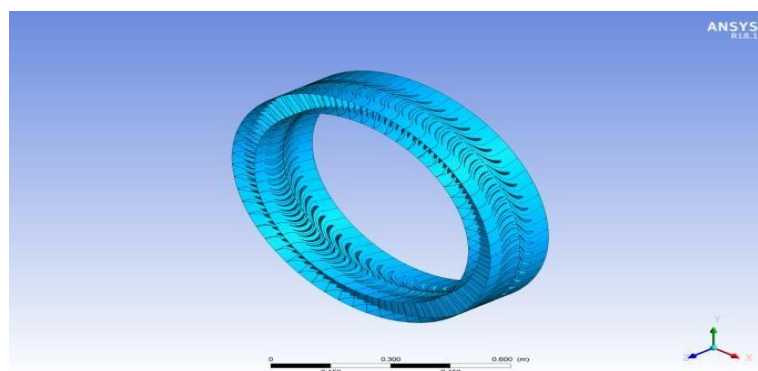
Turning pressurized steam into rotational motion is the job of a steam turbine, a mechanical apparatus. Approximately 80% of the world's electrical power comes from turbines, which are ideal for this purpose since they provide rotational motion, which can then be utilized to power electrical generators. steam turbines are the primary means of power generating

globally. Aerodynamic performance is a major factor in determining a turbine's efficiency. As a result, higher total turbine efficiency has resulted from decades of work refining the design of stators' and rotors' blade profiles. The energy in the steam is transformed into motion when it expands via a nozzle, which is a specially designed tube. The resulting jet is a high-velocity stream with a high tangential component. Then,

by angling the jet correctly and directing it against curved blades attached to a spinning disc, some of the kinetic energy and, sometimes, some of the pressure energy are transformed into mechanical energy. In a perfect steam turbine, the entropy of the steam going into the turbine is the same as the entropy going out, a phenomenon known as an isentropic or constant entropy process. It is common practice to use the isentropic efficiency for expansion process to characterize the steam turbine's efficiency. While there is no such thing as a "isentropic" steam turbine, average isentropic efficiencies of 70% to 90% are achievable depending on the turbine's use. Turbine efficiency drops and blades physically wear down when steam contains water droplets. Hence, the steam's dryness fraction at the turbine's exhaust must be more than or equal to 0.9.

A subfield of fluid dynamics known as computational fluid dynamics (CFD) focuses on finding solutions to fluid flow issues via the use of numerical techniques and algorithms. The main benefit of computational fluid dynamics (CFD) analysis is the reduction of time

and expense associated with predicting the model's behavior in a real-world setting. It is important to review and evaluate the findings of the CFD study prior to accepting the model. The commercial CFD codes that are accessible are ANSYS Fluent and ANSYS CFX. The primary distinction is in the manner in which the flow equations and solution techniques are integrated by the solvers. CFX discretizes the domain using components with finite volumes. On the other hand, Fluent makes use of limited volumes. The conservation of flow quantities is guaranteed by the fact that they are both control volume based solvers. The research utilizes ANSYS CFX to conduct the computational fluid dynamics (CFD) analysis of the steam turbine stage. If you're having trouble with your turbo gear, try using CFX—it's very user-friendly software. During the turbine stage, the stator and rotor interfaces are automatically constructed. In post processing, a full report is provided about the turbine and nozzle performance, including metrics like power, efficiency, mass flow rate, mach number, and more.



1.1. Modelling Turbine Stage in CFX

In general all turbo machinery problems are solved using rotational periodicity and hence a single or double blade passage in the stator and rotor are only modeled to reduce the overall computing time. In this paper single axial turbine stage is modeled. The stator has 60 blades and rotor has 58 blades. The stator or rotor fluid domain consists of various boundaries for inlet, outlet, hub, shroud and other periodic surfaces. Initially CAD drawings for both the



fluid domains are imported into Ansys Turbo grid software and meshing compatible with CFX solver is generated.

The working fluid used in the paper is superheated steam at 2 bar and 3 bar pressure at temperatures 500 and 600K respectively. CFX derives all the thermodynamic properties between inlet and exit pressures through the IAPWS97 database provided as default in the software. Both the fluid domains are imported into the Ansys CFX- pre and the rotational speed is given as input for the rotor. With regard to inlet and outlet boundary conditions CFX has different options, static pressure or velocities and temperature of the fluid at inlet and mass flow rate or average static pressure at outlet. In the present case of the axial turbine the pressure at inlet as well as at outlet are given as boundary conditions. Inlet pressure is 2 bar @500K/ 3 bar @600K and outlet pressure is 1 bar. Thermodynamic properties at inlet as are given below

Inlet condition	2 bar @500K	3 bar @ 600K
Enthalpy, KJ/Kg	2930	3130
Saturation temperature	393K	406K
Degree of superheat	107K	194K
Specific volume, Meter cube/Kg	1.15	0.887

1.2. Turbulence Modelling

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. The strong channeled curvature's and intensive rotations prevalent in turbo machinery resulting in high swirling and secondary flow nictitates choosing appropriate turbulence model for accurate performance prediction. CFX offers many turbulence models and in the present case of axial turbine stage, SST model was chosen as this turbulence model as this model is strongly recommended for turbo machinery problems.

2. RESULTS AND DISCUSSION

The efficiency and power output strongly depend on the inlet conditions of the steam turbine and it is essential to predict the performance in the initial stages of design so as to run the turbine at optimum conditions of maximum efficiency and power output. The turbine at both inlet conditions i.e. 3 bar 600K and 2 bar 500 K is simulated at different speeds i.e. from 6000 to 12000 rpm to obtain optimum conditions.

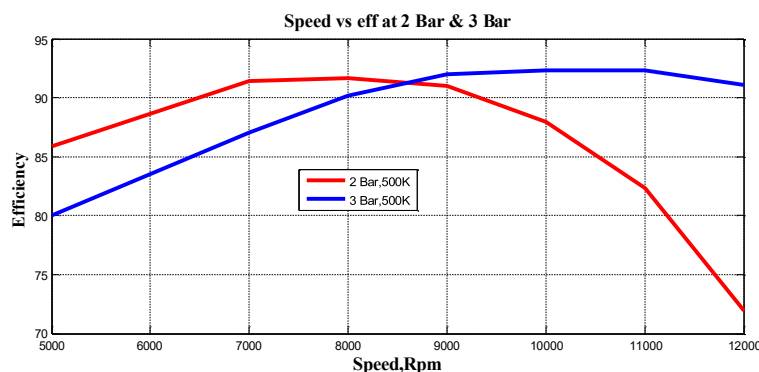


Figure 1.1 Speed vs. Efficiency at 2 Bar & 3 Bar

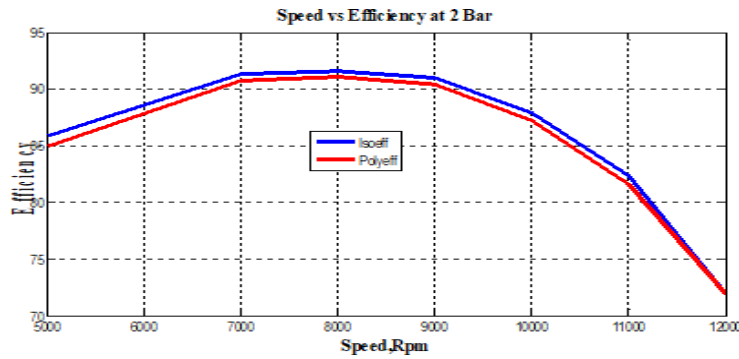


Figure 1.2 Speed vs. Efficiency at 2 Bar

It is seen from above figures 1.1 and 1.2 that both isentropic and polytropic efficiencies show increasing trend initially and there after drops off after reaching maximum value. However at both inlet conditions the maximum efficiencies are above 90% but they do not occur at the same speed i.e. 91% at 8000 rpm for 2 bar while 93% at 10000 rpm at 3 bar. Both polytropic and isentropic efficiencies show similar trend and are almost equal at all speeds as seen in fig 1.2.

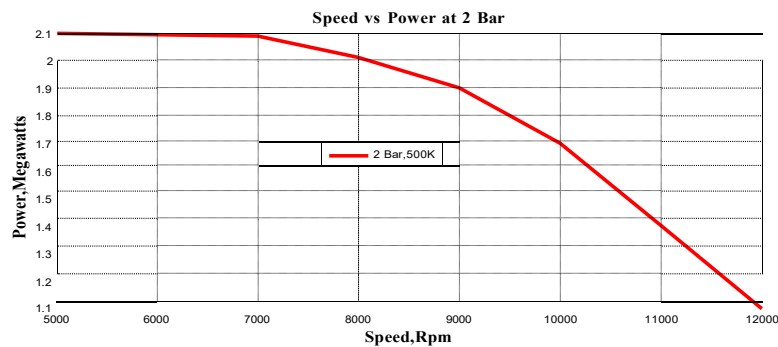


Figure 1.3 Speed vs. Power at 2 Bar

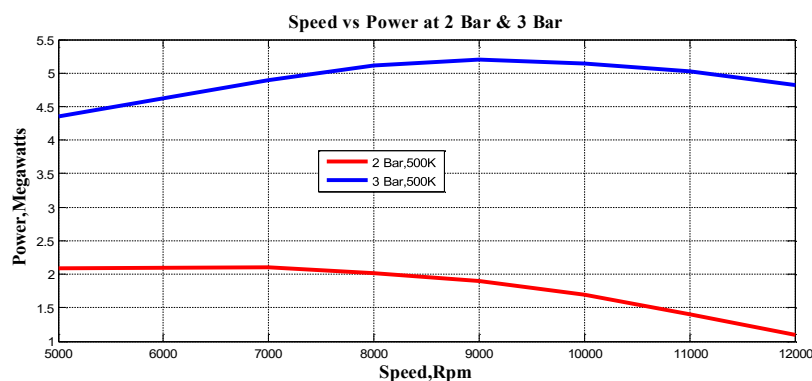


Figure 1.4 Speed vs. Power at 2 Bar & 3 Bar

Power output of the turbine stage depends on the enthalpy drop in the rotor which in turn depends on the rotor speed. As observed in figs 1.3 to 1.4 power output variation with speed shows more or less similar trend with efficiency variation at both inlet conditions. Maximum power produced is more than double at 3 bar i.e. 5 Megawatts at 3 bar and 2 Megawatts at 2bar.

2.1. Velocity distribution in the axial turbine stage

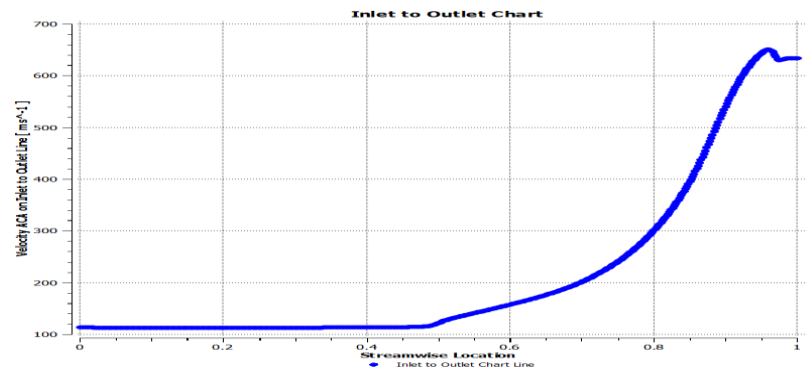


Figure 1.5 Velocity distribution across the Stator

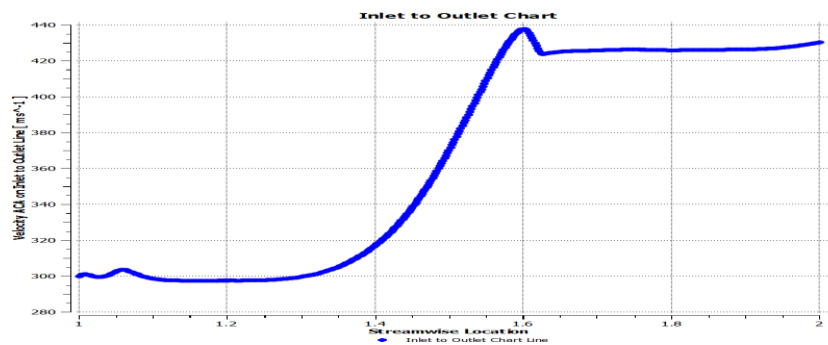


Figure 1.6 Velocity distribution across the rotor

Speed Vs Velocity @ 3 Bar

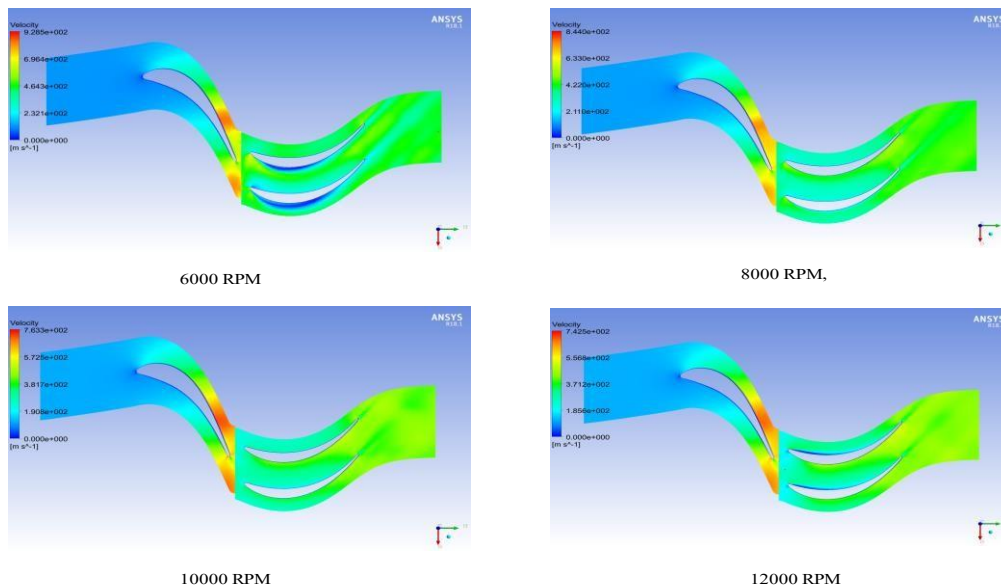


Figure 1.7 Velocity distribution across the Stator & Rotor at different speeds

Figs 1.5 to 1.7 indicate the velocity variation across the stator and rotor. The Kinetic energy increase through the stator is clearly seen in the stator guide vanes since these vanes acts as steam nozzles. The velocity at exit is 650 m/sec while at inlet it is only 100 m/sec. though the exit stator velocity is very high; the rotor inlet velocity is only 300 m/sec and

gradually increases to 430 m/sec due to conversion of part of pressure energy into kinetic energy in the rotor.

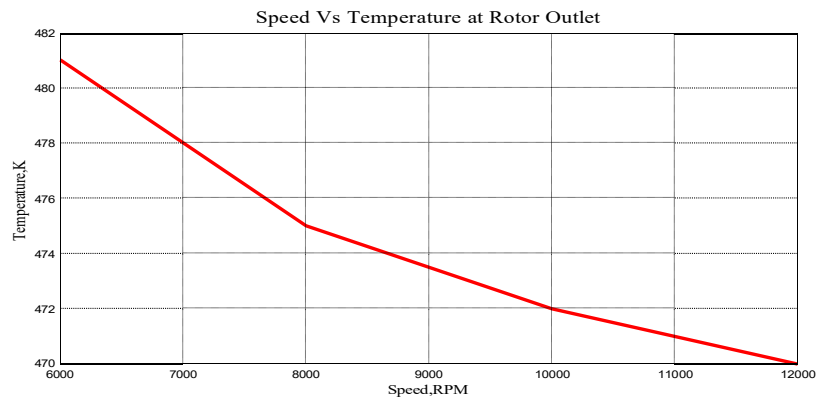


Figure 1.8 Speed vs. Temperature at Rotor Outlet at 3 bar

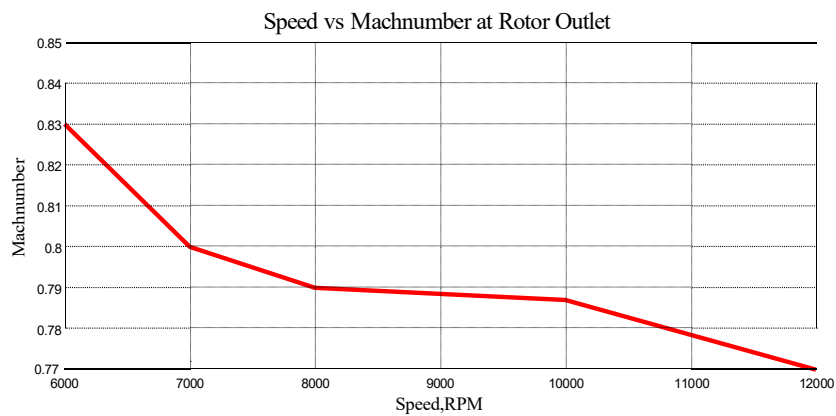


Figure 1.8 Stream wise plot of Relative Mach Numbers at 3 bar, 12000 rpm

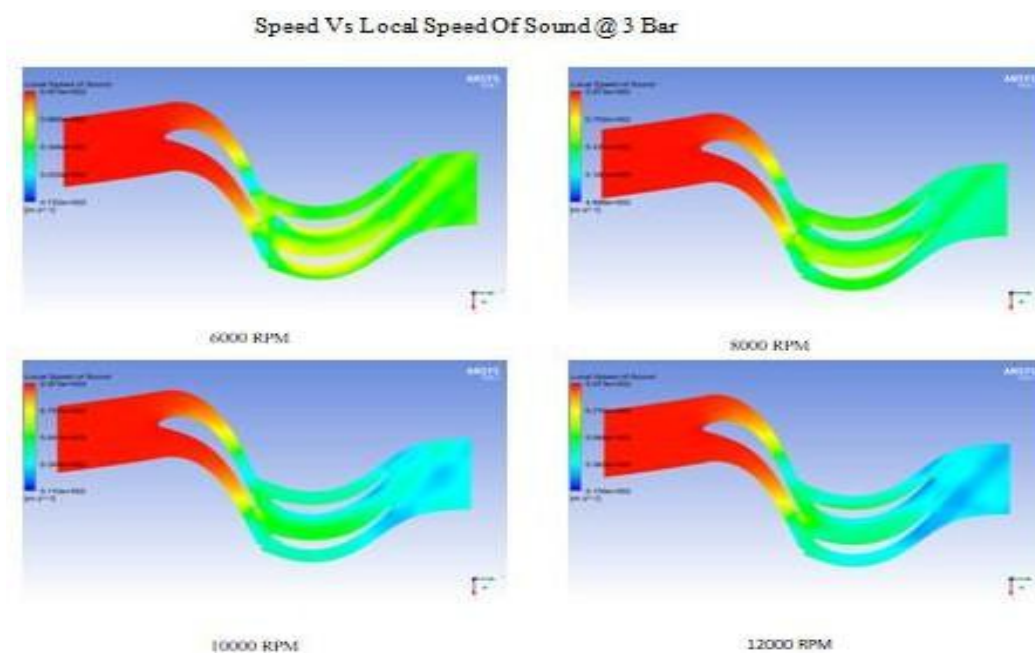


Figure 1.9 Speed vs. Local Speed of Sound @3 Bar

Figs 1.7 to 1.9 shows variation of temperature and relative mach number and local sound speed in the rotor when speed is increased from 6000 to 12000 rpm. Relative Mach number not only depends on the velocity and sonic velocity at the corresponding point, but also on the speed of the rotor. The sonic velocity in turn depends on the temperature and ratio of specific heat at the corresponding point. Fig 1.9 clearly shows that the local speed of sound in the stator fluid domain mostly remains the same at all speeds investigated i.e. from 6000 to 12000 rpm where as the region near rotor out let is different at each speed as evident from different color contours. The temperature plot shows decreasing trend with respect to the rotor speed which in turn cause reduction in sonic velocity and consequently the mach number. The rotor exit temperature is 470K at 3 bar which is more than the saturation temperature corresponding to 3 bar indicating the condition of steam remains superheated with degree of superheat equal to 64 K as against 194K at turbine inlet.

3. CONCLUSIONS

The power generating sector and maritime propulsion are two of steam turbines' most common uses. Thorough simulation studies must be conducted to evaluate the turbine's performance before the final design is frozen. Efficient performance assessment is offered by both Fluent CFD and Ansys CFX, with CFX being seen as more user-friendly when it comes to turbo machinery simulation issues. Here, we focus on a single turbine stage operating at speeds between 6,000 and 12,000 rpm using superheated steam as the working fluid, with pressures of 2 bar 500 K and 3 bar 600 K. Fluid operations have been performed using the IAPWS 97 database. For the two intake circumstances that were taken into consideration, the maximum power output is discovered to occur at distinct

speeds: 5 Megawatts at 10,000 rpm at 3 bar and 2 Megawatts at 8,000 rpm. The stator blade passage experiences a significant increase in kinetic energy, leading to a nozzle efficiency of 94%. As the speed increased, there was a tendency toward lower temperatures and a lower relative Mach number. Under both input circumstances, efficiency levels over 90% were achieved. With a saturation temperature of 3 bar and a rotor exit temperature of 470 K, the steam is still superheated, with a degree of superheat of 64 K, as opposed to 194 K at the turbine intake.

REFERENCES

- [1] V. Boiko and D. I. Maksjuta, Computational Fluid Dynamics for the Optimal Design of High Pressure Steam Turbine Stage, Kharkiv, Ukraine, National Technical University «Kharkiv Polytechnical Institute»
- [2]. The paper "Computational Fluid Dynamics (CFD) Analysis of Intermediate Pressure Steam Turbine" was presented at the ATI2017 conference in Lecce, Italy, from September 6th to the 8th, 2017. The author, Shivakumar Vasmate1, is a member of the International Journal of Mechanical Engineering and Robotics.
- Jurij Bellucci Sazzini Federica Roberto Rubechini Proceedings of the 2013 ASME Turbo Expo: Turbine Technical Conference and Exposition GT2013, June 3-7, San Antonio, Texas, USA, Andrea Arnone, "Using CFD to Enhance The Preliminary Design of High-Pressure Steam Turbines."
- [4] "Computational Fluid Dynamics, the basics with applications" by John D. Anderson Jr., published by Inc. in New York, San Francisco, and Auckland.



[5] Tata McGraw Hill, 2006, Dr. R. K. Bansal's "Fluid Mechanics and Hydraulic Machines"

[6] Journal of Turbomachinery, vol. 130/021016, April 2008; C.W. Haldeman and R.M. Mathison, Aerodynamic and Heat Flux Measurements in a Single-Stage Fully Cooled Turbine - Part II. [7] X. Yan and T. Takinuka conduct computational fluid dynamics (CFD) analysis and test the aerodynamic design model of a multistage axial helium compressor in the Journal of Turbomachinery, an ASME publication.

This is a reference to an article published in July 2008 by Arun K. Saha and Sumanta Acharya in the Journal of Turbomachinery, volume 130, issue 3, which deals with computational models of turbulent flow and heat transfer via a three-dimensional nonaxisymmetric blade passage. [9] J.Horloc's 1957 ASME article "The Thermodynamic Efficiency of the Field Cycle" (Volume 57, No. 44).